

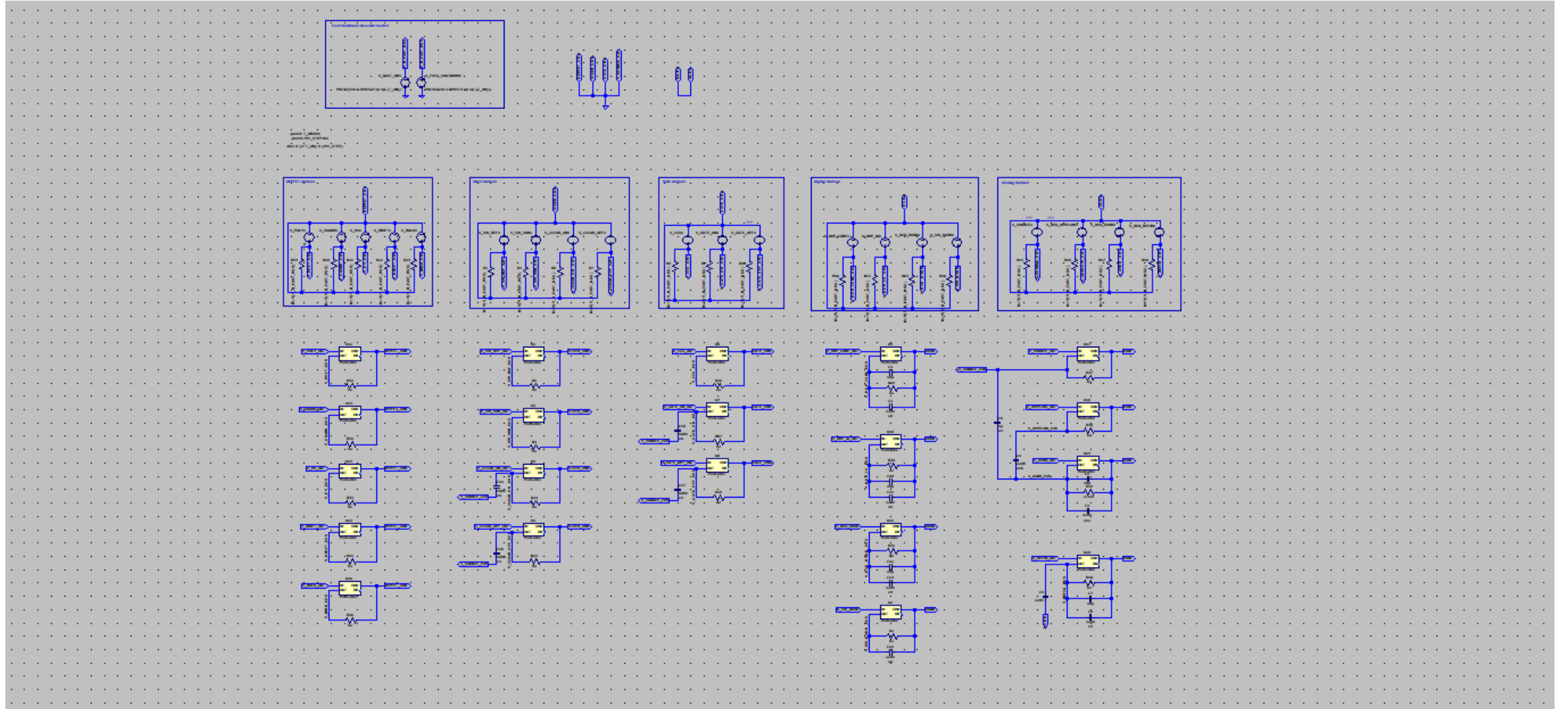
# PS FAST SHUTDOWN SPICE SIMULATION

Matthias Hoek

# STARTING POINT

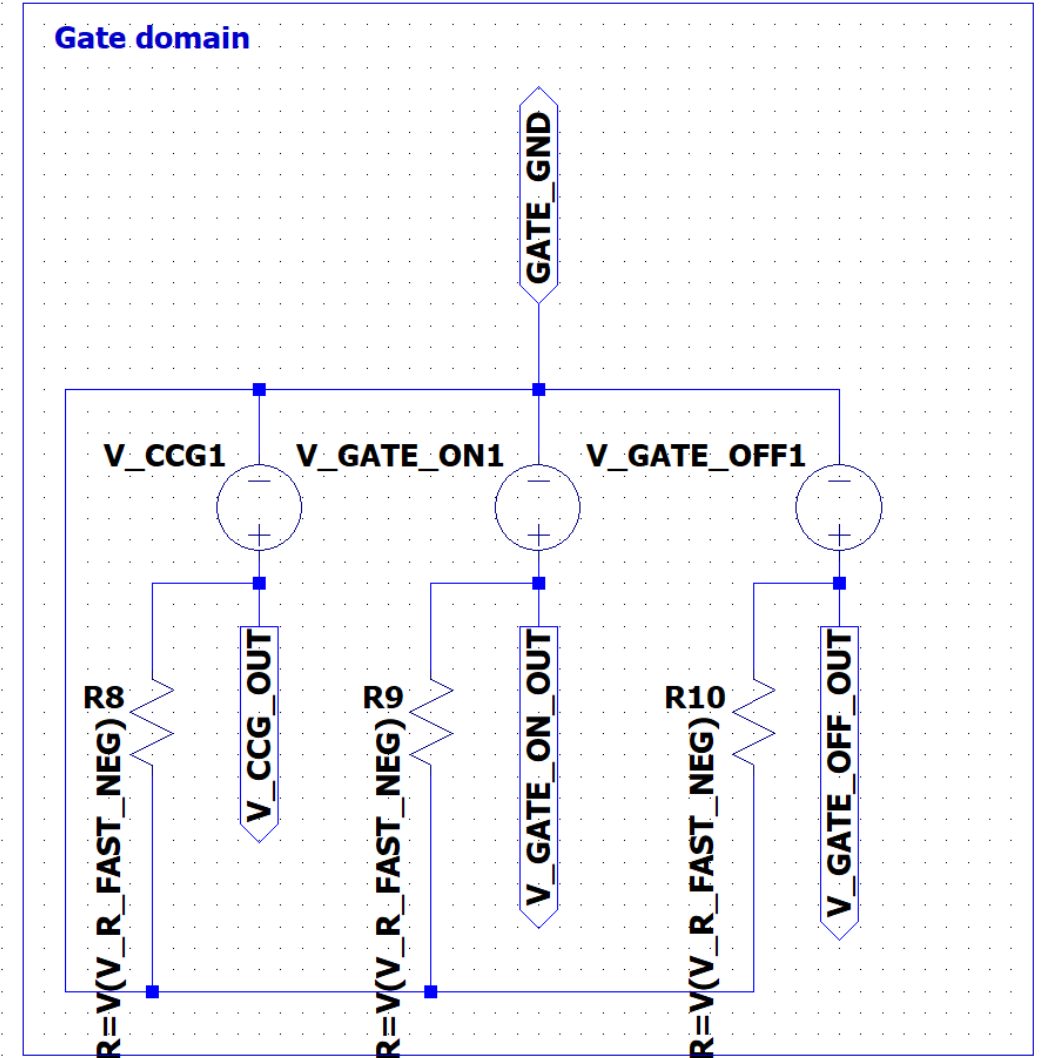
- First effort by Marike and Philip
  - Different PS domains implemented
  - ASICs represented by resistor only
  - No matrix
- Using the cable model from Michael
  - Fast shutdown resistor included ( $R_{\text{fast}}=150\Omega$ )
- Using LTspiceXVII
  - <https://www.analog.com/en/design-center/design-tools-and-calculators/ltspice-simulator.html>

# LTSPICE SIMULATION



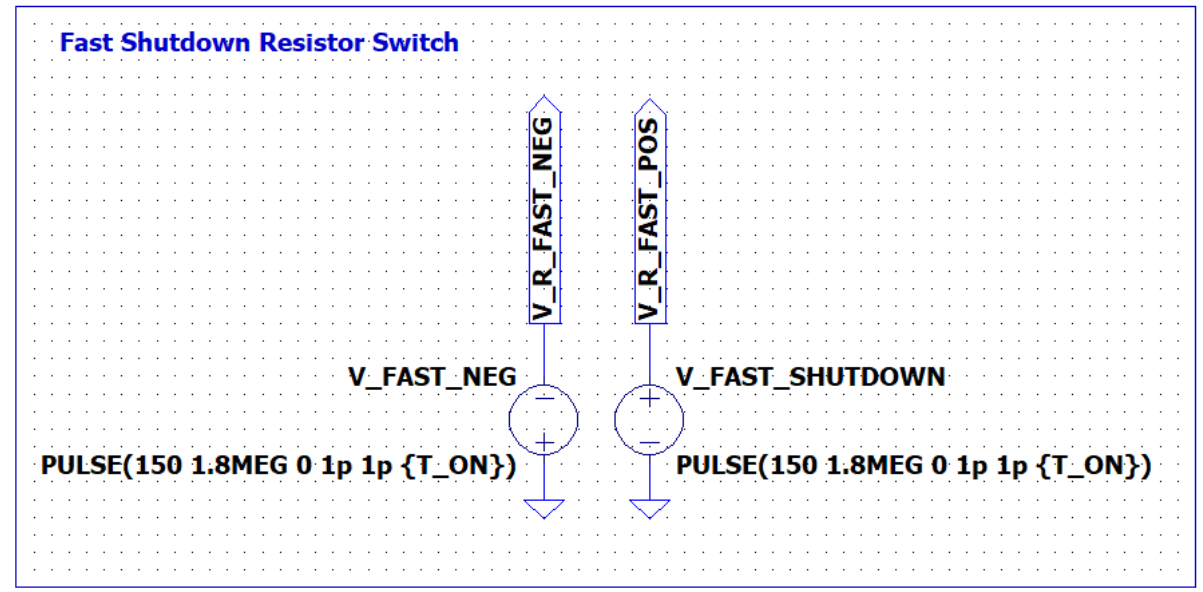
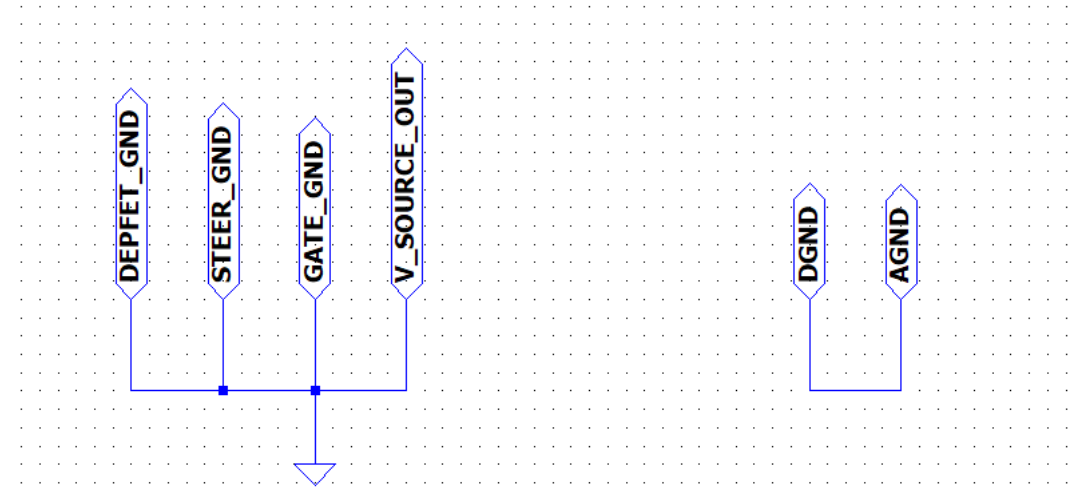
# LTSPICE SIMULATION

- Different PS domains implemented
  - DEPFET
  - Steer
  - Gate
  - Digital
  - Analog
- Using labels to keep layout clear
- Rise and fall time set to 1ps
- Duration can be set via parameter
  - Currently 1ms



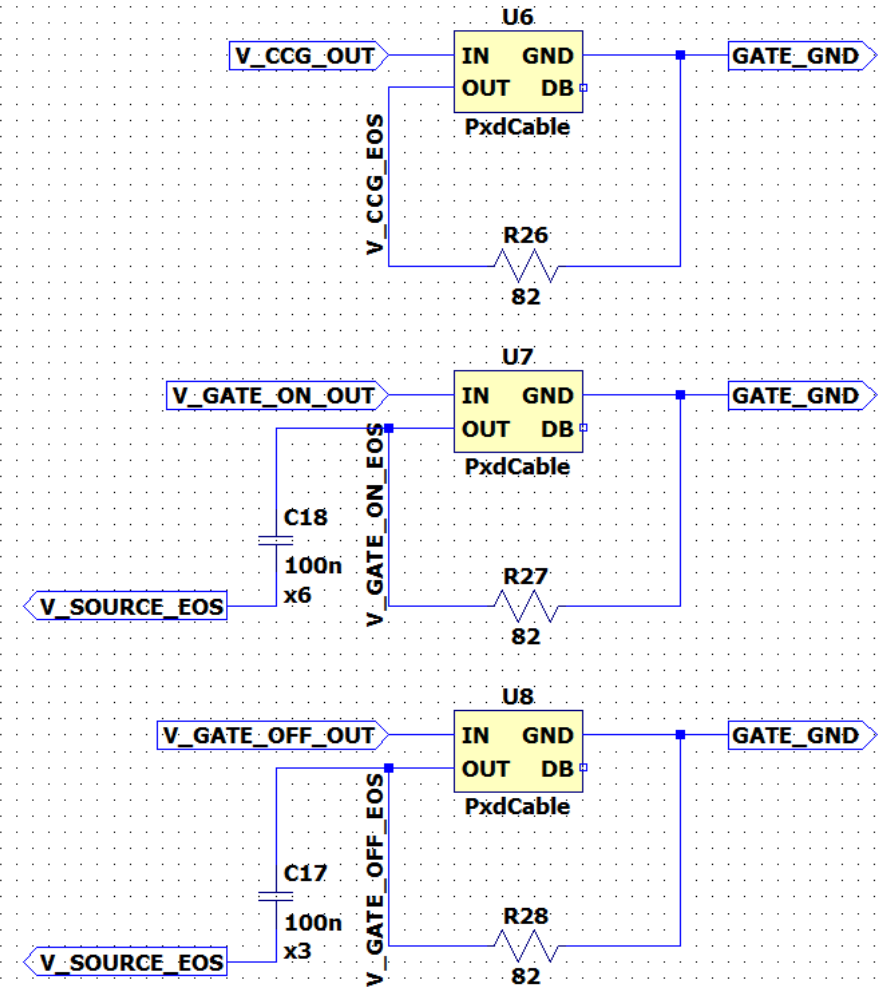
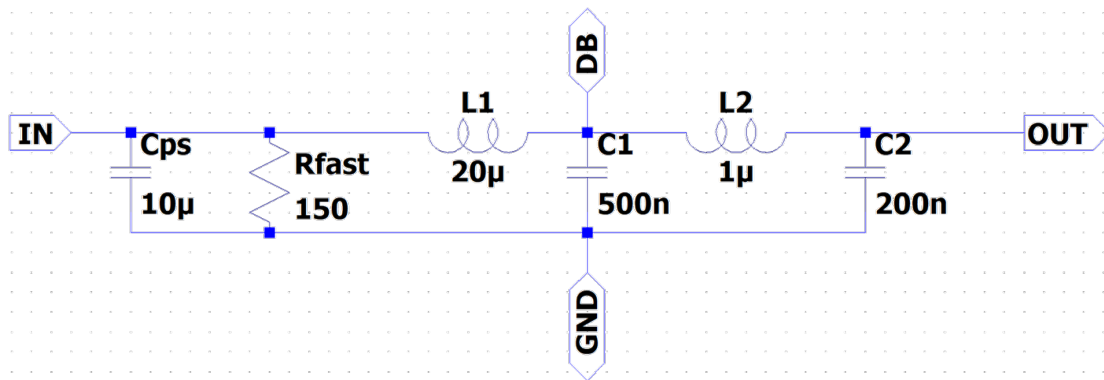
# LTSPICE SIMULATION

- Grounding scheme
- Fast Shutdown
  - Resistor switchable
    - 1.8M $\Omega$  to 150 $\Omega$
  - Rise and fall time 1ps
  - Same parameter for switching as for voltage sources



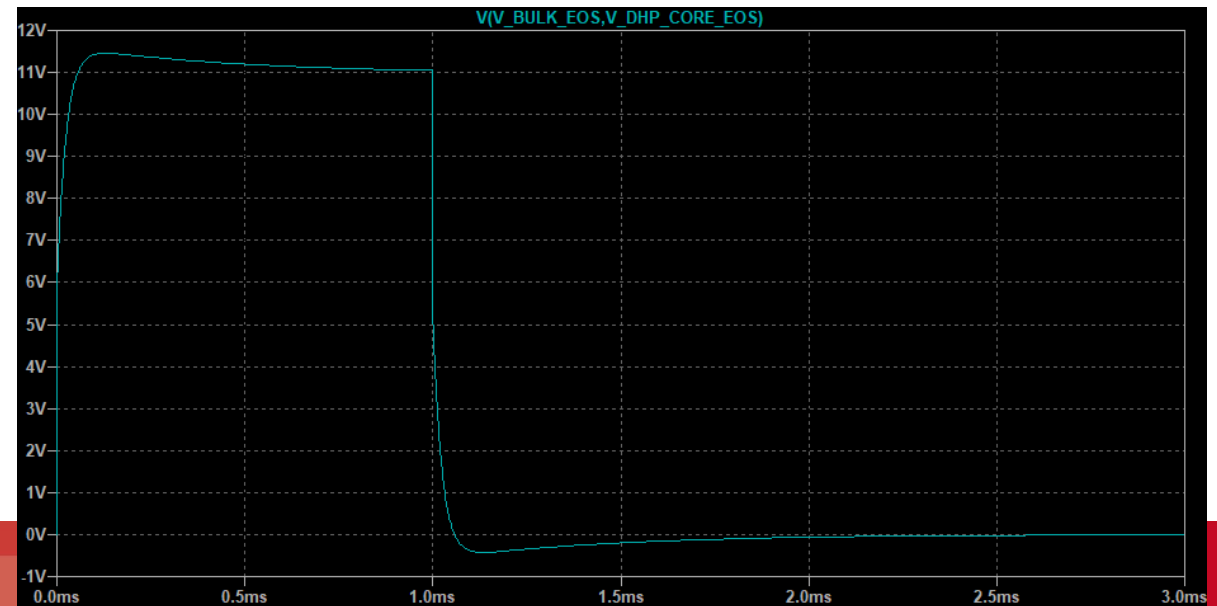
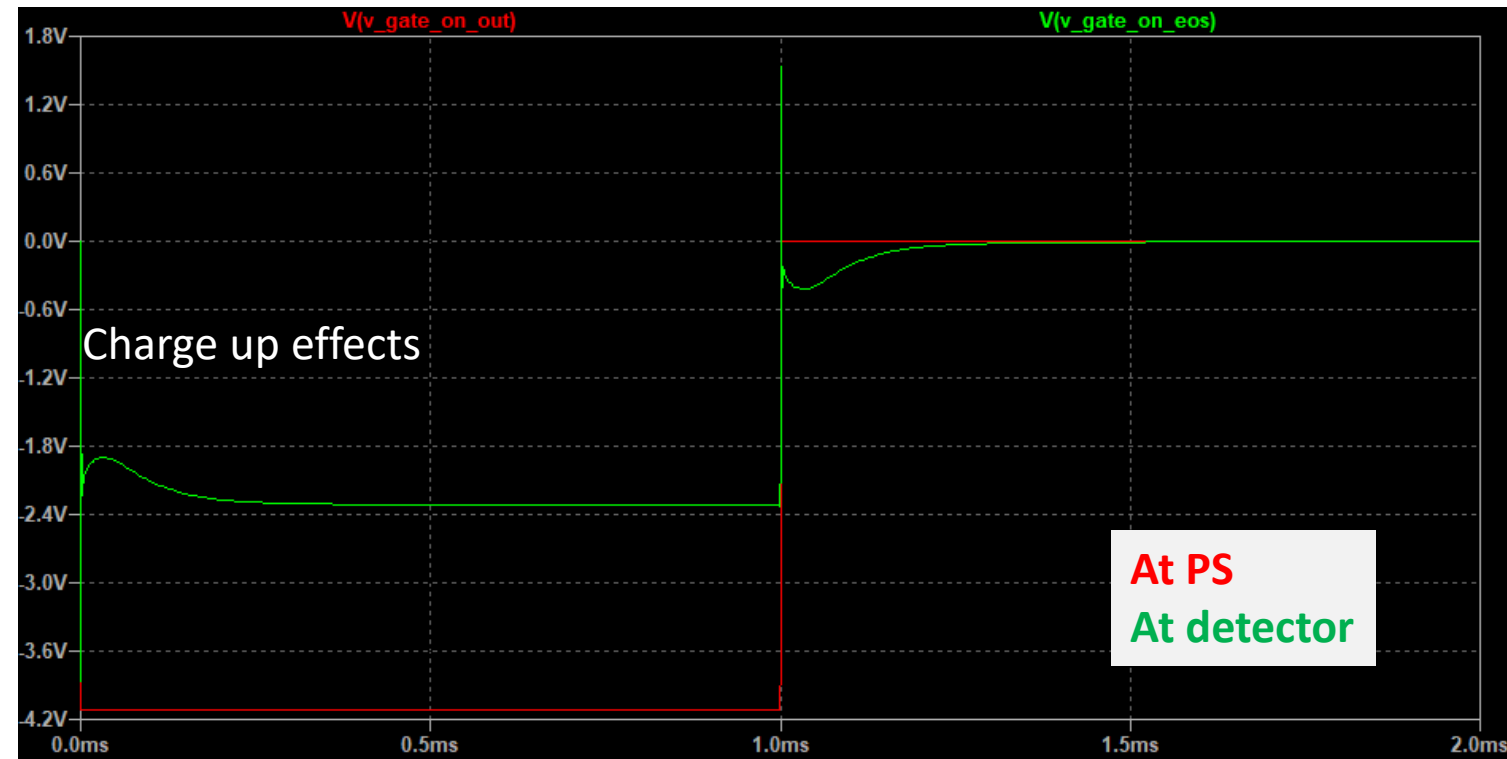
# LTSPICE SIMULATION

- ASICs represented by resistors only
  - Same value for all ( $82\Omega$ )
- Cable model based on Michael's advice
  - Two parts: PS to Dockbox (DB) and DB to detector



# FIRST RESULTS

- Runs w/o errors!
- Capacitors initially not charged up
- Measurements between arbitrary points possible
- Rise and fall time
  - $t_{\text{fall, sw sub}} \approx 53\mu\text{s}$
- Automate via measurement file



# SHORTCOMINGS & IMPROVEMENTS

- ASIC model too simple
- Matrix missing
- No induction in cables
- Remove  $R_{fast}$  from cable
  - Is now included in PS part
- Use more parametric values
- Use Python interface to LTSPICE